

# Stress Analysis of a Roller Chain Sprocket Assembly

This model is licensed under the COMSOL Software License Agreement 6.0. All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

# Introduction

Chain drives are used for transmitting power from one shaft to another, located at some distance. This example demonstrates how to model a chain sprocket assembly in 3D. The geometry consists of a roller chain wrapped around two sprockets. All components are assumed to be elastic. The system dynamics is initiated by an angular velocity prescribed at one of the sprockets. The motion is transmitted via chain links to the second sprocket which is subjected to an external, counteracting torque.

The chain sprocket assembly geometry is created using built-in geometry parts. The **Chain Drive** node in the Multibody Dynamics interface is used for setting up the entire model. A transient analysis is performed to understand the load path, the contact forces and the stress distribution in various components of the assembly.

# Model Definition

The geometry consists of an assembly of a chain wrapped around two sprockets in 3D, as shown in Figure 1. The chain is assumed to be of roller type, and composed of a number of link plates. A typical roller chain has two different types of link plates: roller plates and pin plates.



Figure 1: Geometry of a roller chain sprocket assembly.

Figure 2 shows an exploded view of a sample link connection. A roller plate is made of two hollow cylinders connected to side plates. A pin plate consists of two solid cylinders and

two side plates. The link connection permits relative rotation between the two link plates, and the relative rotation is modeled using hinged joints. Elastic bushings are sometimes used to reduce vibrations in the system, but they are not included in this analysis.



Figure 2: Exploded view of the chain link connection. The connection between links is formed by inserting the pin plates into the hollow roller plate cylinders.

# CHAIN AND SPROCKET PARAMETERS

The distance between two adjacent links is called *pitch*. In this example, the pitch is 0.25 in. All other geometric dimensions of link plates are taken as standard and parameterized as a function of the pitch. Table 1 lists the chain parameters.

The two sprockets have 30 and 15 teeth, respectively. They are located at a distance of 3 in. Each sprocket has a bore at its center, which enables mounting of the sprocket on other mechanical components such as shafts. Table 2 lists the sprocket parameters.

PARAMETER	NAME	VALUE
Pitch	р	0.25 in
Roller diameter to pitch ratio	Dr	0.52
Pin diameter to pitch ratio	Dp	0.362
Bushing diameter to pitch ratio	Db	0.45
Minimum link plate width to roller diameter ratio	Wl	0.6
Thickness to pitch ratio of roller plates (r) and pin plates (p)	tr,tp	0.1

TABLE I: CHAIN PARAMETERS

TABLE 2: SPROCKET PARAMETERS

PARAMETER	NAME	VALUE
Pitch	р	0.25 in
Number of teeth, first sprocket	n1	30
Number of teeth, second sprocket	n2	15
Sprockets center distance	cdx	3 in
Sprocket width to pitch ratio	W	0.5
Sprocket clearance to pitch ratio	clrsp	0.05
Bore diameter to pitch diameter ratio of both sprockets	Dbr	0.2
Hub diameter to pitch diameter ratio of both sprockets	Dh	0.4
Hub width to pitch ratio for upside and downside of both sprockets	Wh	0.1

# MATERIALS

Both the chain and the sprockets are made of structural steel. All components are assumed to be elastic.

## CHAIN DRIVE

The assembly of the chain and the sprockets is modeled using the **Chain Drive** node. For a selected geometry part, this node automatically creates attachments, hinge joints, and contact pairs that can be used to analyze the roller chain assembly.

# SELECTION SETTINGS

For automatic generation of physics nodes, you need to select a geometry of the chain sprocket assembly from the **Part Library** of COMSOL Multiphysics. You can also create your own geometry; however, the geometry requires domain and boundary selections to be used as inputs for automatic creation of other physics nodes. In this example, the following selections are used to create physics nodes using the **Chain Drive** node.

- Pin Outer Boundaries: This is a boundary selection containing the outer cylindrical surfaces of all the pin plates. The selection is used for creating **Attachment** nodes for each of the pin plates.
- Roller Inner Boundaries: This is a boundary selection containing inner cylindrical surfaces of all the roller plates. This selection is used for creating **Attachment** nodes for each of the roller plates. The selection coincides geometrically with the Pin Outer

Boundaries (Figure 3). The attachments for the pin and roller plates are used as **Source** and **Destination** in **Hinge Joint** nodes.



Figure 3: The Pin Outer Boundaries (also Roller Inner Boundaries) selection for creating attachments.

- Roller Outer Boundaries: This boundary selection contains the outer cylindrical surfaces of all the roller plates. The selection is used for creating **Contact Pair** nodes for modeling contact between links and sprockets, as shown in Figure 4.
- Sprocket Outer Boundaries: This boundary selection contains the outer surfaces of the sprockets. The selection is used for creating a **Contact Pair** for modeling contact between links and sprockets as shown in Figure 4.



Figure 4: Contact between the Roller Outer Boundaries and Sprocket Outer Boundaries.

• Sprocket Inner Boundaries: This boundary selection contains the inner surfaces of the sprockets, as shown in Figure 5. It is used for creating attachments and hinge joints for mounting the sprockets onto shafts.



Figure 5: The Sprocket Inner Boundaries selection for creating attachments and hinge joints to mount the sprockets onto shafts.

Additional details about **Chain Drive** functionality can be found in the *Multibody Dynamics Module User's Guide*.

# BOUNDARY CONDITIONS

An angular velocity of 1 rad/s is prescribed on the left sprocket, which acts as a driver for the mechanism. On the driven sprocket, a counteracting torque of 0.01 Nm is applied.

A swept mesh is created for both the sprockets and the chain links, as shown in Figure 6



Figure 6: Swept mesh for the sprockets and the chain links.

## STUDY

A Time Dependent study is used to analyze the chain drive. The analysis is performed for a duration of 0.1 s. The load path, contact forces, and stress distribution, in various components of the chain drive, are analyzed.

Results and Discussion

Figure 7 shows the total displacement of the sprockets and the chain links surfaces, at t = 0.1 s.



Figure 7: The displacement of the sprockets and the chain links, at t = 0.1 s.

The stress state in the chain drive is shown in Figure 8. The figure shows that the links at the top side are under tension and experience higher stresses compared to the links at the bottom. The stresses in the sprocket are shown separately in Figure 9. To understand the local stress distribution in chain links that are under tension, two sample links from the top side are used. Figure 10 shows the von Mises stress distribution for these.

The contact dynamics between the sprockets and the chain outer boundaries is modeled using the penalty method. This method is based on a penalty factor, where penetration of structural components in contact is prevented by introducing a spring stiffness. Figure 11 shows the distribution of contact pressure on the outer boundaries of the driver sprocket.



Figure 8: The von Mises stress distribution in the chain drive, at t = 0.1 s



Figure 9: The von Mises stress distribution in the sprockets, at t = 0.1 s



Figure 10: The von Mises stress distribution for chain links under tension, at t = 0.1 s.Time=0.1 sSurface: Contact pressure  $(N/m^2)$ 



Figure 11: The distribution of contact pressure on outer surfaces of the driver sprocket, at t = 0.1 s.

In order to understand the variation of joint forces over time, four links in the chain are examined. These links are taken from four different locations along the chain: The first link is initially situated at the top tension side of the chain. The second and third links are initially in contact with the sprocket outer boundaries. The initial position of the fourth link is at the bottom slack side of the chain. Figure 12 plots the joint forces for these links, as a function of time.



Figure 12: Link joint force as a function of time.

# Notes About the COMSOL Implementation

- To build a chain drive system geometry, you can import a roller chain sprocket assembly part from the **Part Library**, and customize it by changing its input parameters.
- The Chain Drive node operates on the geometry in the assembly state.
- The **Chain Drive** node creates new physics nodes from selections available in the geometry. If you are using a part imported from the **Part Library**, select the check box in **Geometry** to keep noncontributing selections. If you are building a geometry of a roller chain and sprocket assembly, you also need to create appropriate selections for the **Chain Drive** node to operate.
- When one or more selection inputs of the **Chain Drive** node change, the selections of the physics nodes created by **Chain Drive** node also change. Hence, these nodes have to be deleted and recreated. This is indicated by a warning node appearing under the **Chain Drive** node. In that case, you need to press the **Create Links and Joints** button. This will

automatically create new groups of physics nodes in accordance with the changed selection inputs.

**Application Library path:** Multibody\_Dynamics\_Module/Tutorials, \_Transmission/elastic\_roller\_chain

# Modeling Instructions

From the File menu, choose New.

## NEW

In the New window, click 🙆 Model Wizard.

# MODEL WIZARD

- I In the Model Wizard window, click 间 3D.
- 2 In the Select Physics tree, select Structural Mechanics>Multibody Dynamics (mbd).
- 3 Click Add.
- 4 Click  $\bigcirc$  Study.
- 5 In the Select Study tree, select General Studies>Time Dependent.
- 6 Click M Done.

# GLOBAL DEFINITIONS

Parameters 1

- I In the Model Builder window, under Global Definitions click Parameters I.
- 2 In the Settings window for Parameters, locate the Parameters section.
- **3** In the table, enter the following settings:

Name	Expression	Value	Description
omega	1[rad/s]	l rad/s	Angular velocity of drive shaft
T_ext	0.01[N*m]	0.01 N·m	External torque
рс	1.5e8[N/m^3]	1.5E8 N/m <sup>3</sup>	Penalty factor

#### DEFINITIONS

Step I (step I)

- I In the Home toolbar, click f(X) Functions and choose Local>Step.
- 2 In the Settings window for Step, locate the Parameters section.
- 3 In the Location text field, type 0.01.
- 4 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type 0.02.

Step 2 (step 2)

- I Right-click Step I (step I) and choose Duplicate.
- 2 In the Settings window for Step, locate the Parameters section.
- 3 In the Location text field, type 0.03.

## PART LIBRARIES

- I In the Home toolbar, click 📑 Windows and choose Part Libraries.
- 2 In the Part Libraries window, select Multibody Dynamics Module>3D>Roller Chains> roller\_chain\_sprocket\_assembly in the tree.
- **3** Click  **Add to Geometry**.
- 4 In the Select Part Variant dialog box, select Specify sprocket center distance in the Select part variant list.
- 5 Click OK.
- GEOMETRY I

#### Roller Chain Sprocket I (pil)

- I In the Model Builder window, under Component I (compl)>Geometry I click Roller Chain Sprocket I (pil).
- 2 In the Settings window for Part Instance, locate the Selection Settings section.
- **3** Select the Keep noncontributing selections check box.
- 4 Click 틤 Build Selected.

#### Form Union (fin)

- I In the Model Builder window, under Component I (compl)>Geometry I click Form Union (fin).
- 2 In the Settings window for Form Union/Assembly, locate the Form Union/Assembly section.
- 3 From the Action list, choose Form an assembly.

- 4 Clear the **Create pairs** check box.
- 5 Click 틤 Build Selected.

#### ADD MATERIAL

- I In the Home toolbar, click 🙀 Add Material to open the Add Material window.
- 2 Go to the Add Material window.
- 3 In the tree, select Built-in>Structural steel.
- 4 Click Add to Component in the window toolbar.
- 5 In the Home toolbar, click 🙀 Add Material to close the Add Material window.

# MULTIBODY DYNAMICS (MBD)

#### Chain Drive I

- I In the Model Builder window, under Component I (compl) right-click Multibody Dynamics (mbd) and choose Chain Drive.
- 2 In the Settings window for Chain Drive, locate the Chain Settings section.
- 3 From the Link type list, choose Elastic.
- 4 Locate the Joint Settings section. From the Attachment type list, choose Flexible.
- 5 Click Create Links and Joints in the window toolbar.

# Hinge Joint: Sprocket: I (cdr1)

In the Model Builder window, expand the Component I (comp1)> Multibody Dynamics (mbd)>cdr1: Sprocket node, then click Hinge Joint: Sprocket: I (cdr1).

# Prescribed Motion 1

- I In the Physics toolbar, click 📃 Attributes and choose Prescribed Motion.
- **2** In the **Settings** window for **Prescribed Motion**, locate the **Prescribed Rotational Motion** section.
- 3 From the Prescribed motion through list, choose Angular velocity.
- 4 In the  $\omega_p$  text field, type omega\*step1(t[1/s]).

# Hinge Joint: Sprocket: 2 (cdr1)

In the Model Builder window, under Component I (compl)>Multibody Dynamics (mbd)> cdrl: Sprocket click Hinge Joint: Sprocket: 2 (cdrl).

#### Applied Force and Moment I

- I In the Physics toolbar, click 📃 Attributes and choose Applied Force and Moment.
- 2 In the Settings window for Applied Force and Moment, locate the Applied On section.

- **3** From the list, choose **Joint**.
- 4 Locate the Applied Force and Moment section. In the *M* text field, type -T\_ext\* step2(t[1/s]).

Contact: Sprocket-Roller (cdr1)

- I In the Model Builder window, expand the Component I (comp1)> Multibody Dynamics (mbd)>cdr1: Contact node, then click Contact: Sprocket-Roller (cdr1).
- 2 In the Settings window for Contact, locate the Contact Pressure Penalty Factor section.
- 3 From the Penalty factor control list, choose User defined.
- **4** In the  $p_n$  text field, type pc.

# DEFINITIONS

## Variables I

- I In the Model Builder window, under Component I (compl) right-click Definitions and choose Variables.
- 2 In the Settings window for Variables, locate the Variables section.

3	In the	table.	enter	the	foll	owing	settings:
•	in the	cuore,	circor	une	10110	o ming	occurry.

Name	Expression	Unit	Description
Flj_32	sqrt(mbd.cdr1hgj32.Fx^ 2+mbd.cdr1hgj32.Fy^2)	N	Link joint force, joint 32
Flj_8	<pre>sqrt(mbd.cdr1hgj8.Fx^2 +mbd.cdr1hgj8.Fy^2)</pre>	N	Link joint force, joint 8
Flj_1	<pre>sqrt(mbd.cdr1hgj1.Fx^2 +mbd.cdr1hgj1.Fy^2)</pre>	Ν	Link joint force, joint 1
Flj_33	sqrt(mbd.cdr1hgj33.Fx^ 2+mbd.cdr1hgj33.Fy^2)	N	Link joint force, joint 33

# MESH I

Swept I

- I In the Model Builder window, expand the Mesh I node.
- 2 Right-click Component I (compl)>Mesh I and choose Swept.

# Size

- I In the Settings window for Size, locate the Element Size section.
- 2 From the Predefined list, choose Finer.

#### Swept I

- I In the Model Builder window, click Swept I.
- 2 In the Settings window for Swept, click to expand the Sweep Method section.
- 3 From the Face meshing method list, choose Triangular (generate prisms).
- 4 Click 📗 Build All.

#### STUDY I

- Step 1: Time Dependent
- I In the Model Builder window, under Study I click Step I: Time Dependent.
- 2 In the Settings window for Time Dependent, locate the Study Settings section.
- 3 In the **Output times** text field, type range(0, 1e-3, 0.1).

## Solution 1 (soll)

- I In the Study toolbar, click The Show Default Solver.
- 2 In the Model Builder window, expand the Solution I (soll) node, then click Time-Dependent Solver I.
- **3** In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.
- 4 From the Steps taken by solver list, choose Intermediate.
- 5 In the Model Builder window, expand the Study I>Solver Configurations> Solution I (soll)>Time-Dependent Solver I node, then click Fully Coupled I.
- **6** In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.
- 7 In the Maximum number of iterations text field, type 5.
- 8 In the **Study** toolbar, click **= Compute**.

Follow the instructions below to plot von Mises stress distribution similar to the one shown in Figure 8.

## RESULTS

von Mises Stress, All

- I Right-click Displacement (mbd) and choose Duplicate.
- 2 In the Settings window for 3D Plot Group, type von Mises Stress, All in the Label text field.

## Surface

- I In the Model Builder window, expand the von Mises Stress, All node, then click Surface.
- 2 In the Settings window for Surface, locate the Expression section.
- 3 In the **Expression** text field, type mbd.mises.
- 4 Click to expand the Range section. Select the Manual color range check box.
- **5** In the **Minimum** text field, type **0**.
- 6 In the Maximum text field, type 1e5.

Follow the instructions below to plot von Mises stress distribution in sprockets as shown in Figure 9.

von Mises Stress, Sprockets

- I In the Model Builder window, right-click von Mises Stress, All and choose Duplicate.
- 2 In the Settings window for 3D Plot Group, type von Mises Stress, Sprockets in the Label text field.
- **3** Click to expand the **Selection** section. From the **Geometric entity level** list, choose **Domain**.
- 4 From the Selection list, choose Sprockets (Roller Chain Sprocket I).
- **5** Select the **Propagate to lower dimensions** check box.
- 6 Locate the Plot Settings section. From the View list, choose New view.

#### Surface

- I In the Model Builder window, expand the von Mises Stress, Sprockets node, then click Surface.
- 2 In the Settings window for Surface, locate the Range section.
- 3 In the Maximum text field, type 2e4.

#### Contour I

- I In the Model Builder window, right-click von Mises Stress, Sprockets and choose Contour.
- 2 In the Settings window for Contour, locate the Expression section.
- **3** In the **Expression** text field, type mbd.mises.
- 4 Click to expand the **Title** section. From the **Title type** list, choose **None**.
- 5 Locate the Coloring and Style section. From the Coloring list, choose Uniform.
- 6 From the Color list, choose White.
- 7 Locate the Levels section. In the Total levels text field, type 10.
- 8 Locate the Coloring and Style section. Clear the Color legend check box.

#### Deformation I

- I Right-click Contour I and choose Deformation.
- 2 In the Settings window for Deformation, locate the Scale section.
- **3** Select the **Scale factor** check box.
- 4 In the associated text field, type 1.
- 5 In the von Mises Stress, Sprockets toolbar, click 🗿 Plot.
- 6 Click the **Click the Go to XY View** button in the **Graphics** toolbar.

Follow the instructions below to plot von Mises stress distribution in two selected links as shown in Figure 10.

von Mises Stress, Links

- I In the Model Builder window, right-click von Mises Stress, Sprockets and choose Duplicate.
- 2 In the Settings window for 3D Plot Group, type von Mises Stress, Links in the Label text field.
- 3 Locate the Selection section. Click 🚺 Clear Selection.
- 4 Select Domains 411-413, 417-421, and 439-454 only.
- 5 Select the Apply to dataset edges check box.
- 6 Locate the Plot Settings section. From the View list, choose New view.

#### Surface

- I In the Model Builder window, expand the von Mises Stress, Links node, then click Surface.
- 2 In the Settings window for Surface, locate the Range section.
- **3** Clear the **Manual color range** check box.
- **4** Click the **V Go to Default View** button in the **Graphics** toolbar.
- **5** Click the **Comextents** button in the **Graphics** toolbar.
- 6 Click 💽 Plot.

Follow the instructions below to plot joint forces in four selected links in the chain as shown in Figure 11.

Contact Pressure, Sprocket-I

- I In the Model Builder window, right-click Displacement (mbd) and choose Duplicate.
- 2 In the Settings window for 3D Plot Group, type Contact Pressure, Sprocket-1 in the Label text field.
- **3** Locate the **Selection** section. From the **Geometric entity level** list, choose **Domain**.

- 4 Locate the Plot Settings section. From the View list, choose New view.
- 5 Locate the Selection section. From the Selection list, chooseFirst Sprocket (Roller Chain Sprocket 1).
- 6 Select the Apply to dataset edges check box.
- 7 Select the Propagate to lower dimensions check box.

#### Surface

- I In the Model Builder window, expand the Contact Pressure, Sprocket-I node, then click Surface.
- 2 In the Settings window for Surface, locate the Expression section.
- 3 In the **Expression** text field, type mbd.Tn.

#### Filter I

- I Right-click Surface and choose Filter.
- 2 In the Settings window for Filter, locate the Element Selection section.
- 3 In the Logical expression for inclusion text field, type mbd.Tn>0.
- **4** In the **Contact Pressure, Sprocket-I** toolbar, click **O** Plot.
- **5** Click the  $\longleftrightarrow$  **Zoom Extents** button in the **Graphics** toolbar.

#### Link Joint Forces

- I In the Home toolbar, click 🚛 Add Plot Group and choose ID Plot Group.
- 2 In the Settings window for ID Plot Group, type Link Joint Forces in the Label text field.
- 3 Click to expand the Title section. From the Title type list, choose Manual.
- 4 In the Title text area, type Link Joint Force, Magnitude.
- 5 Locate the Plot Settings section. Select the y-axis label check box.
- 6 In the associated text field, type Force (N).
- 7 Locate the Legend section. From the Position list, choose Upper left.

### Global I

- I Right-click Link Joint Forces and choose Global.
- 2 In the Settings window for Global, locate the y-Axis Data section.

**3** In the table, enter the following settings:

Expression	Unit	Description
Flj_32	Ν	Link joint force, joint 32
Flj_8	Ν	Link joint force, joint 8
Flj_1	Ν	Link joint force, joint 1
Flj_33	Ν	Link joint force, joint 33

**4** In the Link Joint Forces toolbar, click **I** Plot.

Displacement

- I In the **Results** toolbar, click **Animation** and choose **Player**.
- 2 In the Settings window for Animation, type Displacement in the Label text field.
- 3 Locate the Frames section. In the Number of frames text field, type 50.

von Mises Stress, All

- I Right-click **Displacement** and choose **Duplicate**.
- 2 In the Settings window for Animation, type von Mises Stress, All in the Label text field.
- 3 Locate the Scene section. From the Subject list, choose von Mises Stress, All.

von Mises Stress, Sprockets

- I Right-click von Mises Stress, All and choose Duplicate.
- 2 In the Settings window for Animation, type von Mises Stress, Sprockets in the Label text field.
- 3 Locate the Scene section. From the Subject list, choose von Mises Stress, Sprockets.

von Mises Stress, Links

- I Right-click von Mises Stress, Sprockets and choose Duplicate.
- 2 In the Settings window for Animation, type von Mises Stress, Links in the Label text field.
- 3 Locate the Scene section. From the Subject list, choose von Mises Stress, Links.

Contact Pressure, Sprocket-1

- I Right-click von Mises Stress, Links and choose Duplicate.
- 2 In the Settings window for Animation, type Contact Pressure, Sprocket-1 in the Label text field.
- 3 Locate the Scene section. From the Subject list, choose Contact Pressure, Sprocket-I.

20 | STRESS ANALYSIS OF A ROLLER CHAIN SPROCKET ASSEMBLY